Vehicle thermal safety with THESEUS-FE

Author: Dr. Daniel Koester

Date: September 4, 2014

P+Z Engineering is a member of the ARRK Research & Development Group

Workflow for Underhood Simulations

ANSA or other CAD/meshing software

- **CAD cleanup and meshing:** either CAD geometry description or NASTRAN-style FE meshes

THESEUS-FE

- **Thermal model building:** density, specific heat capacity, and conductivity of all vehicle parts, emissivity of important surfaces

Star-CCM+

- **CFD model building:** air mass flow and temperature from fan and heat exchanger and other inlets

THESEUS-FE Coupler

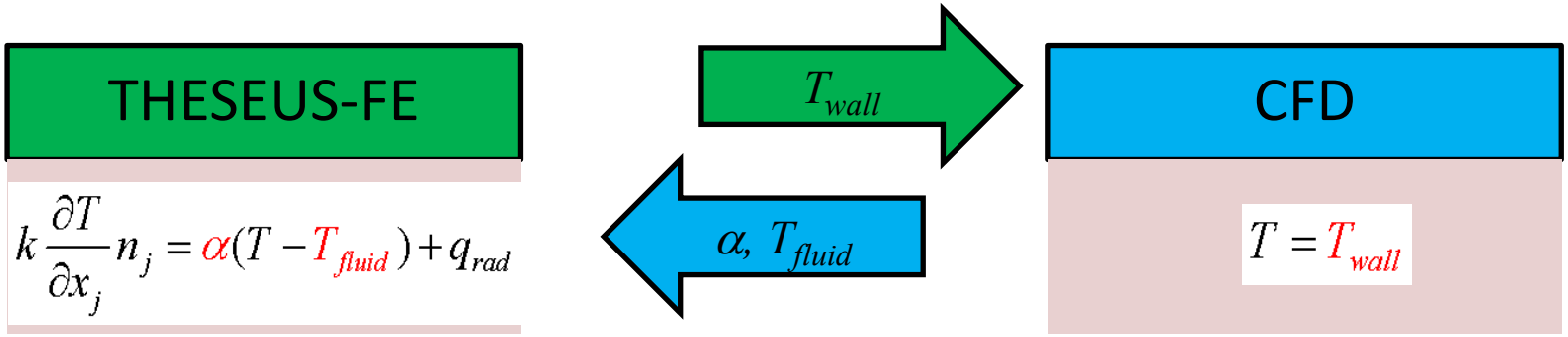
- **Coupled simulation setup and analysis:** identification of coupled interfaces between solid parts and air (engine, gearbox, heat shields, etc.)

THESEUS-FE or other postprocessing software

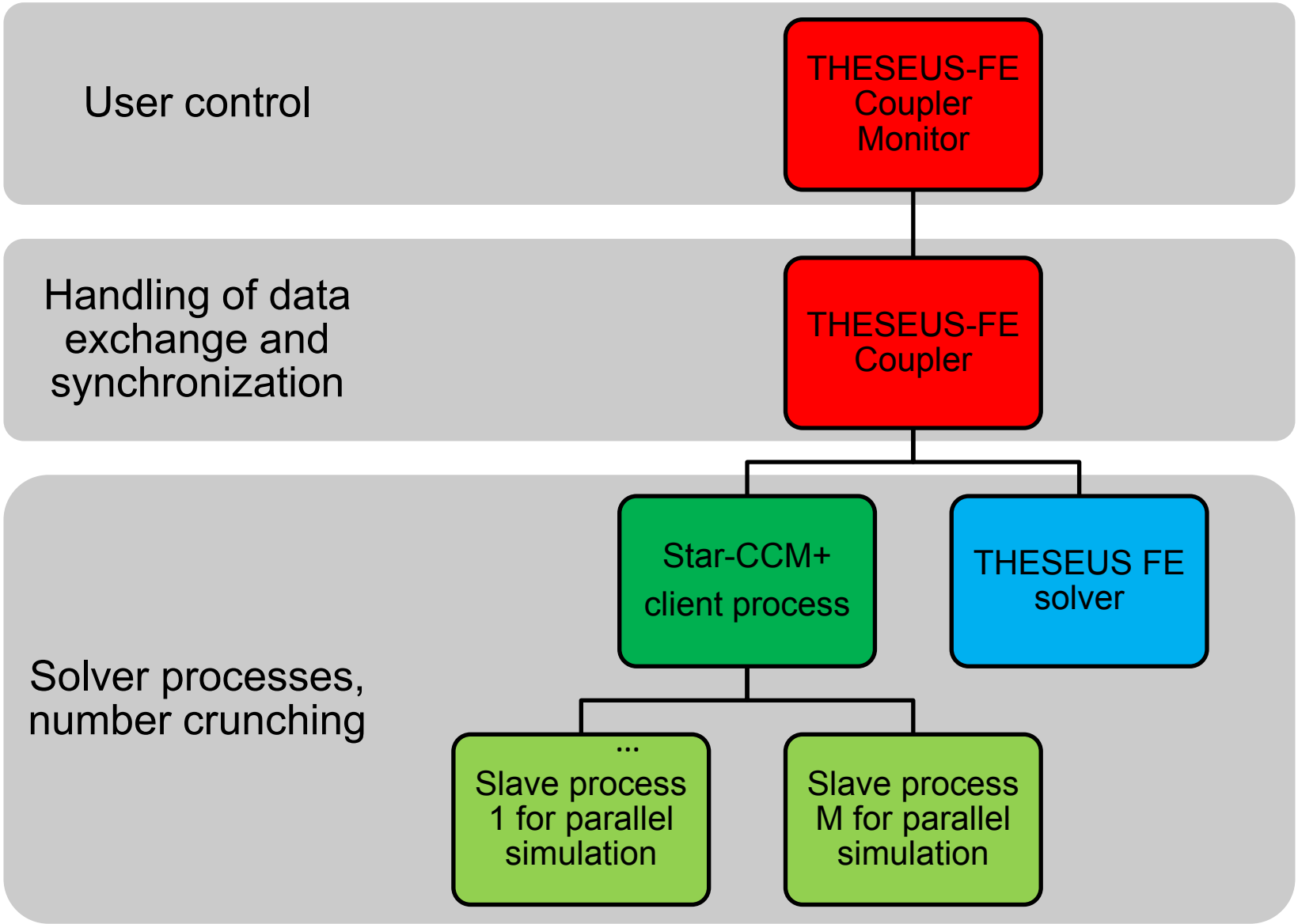
- Identification of hot spots or heat flow balance

Exchange of BC data

The interaction between individual solvers is done through the boundary conditions:



Implementation architecture

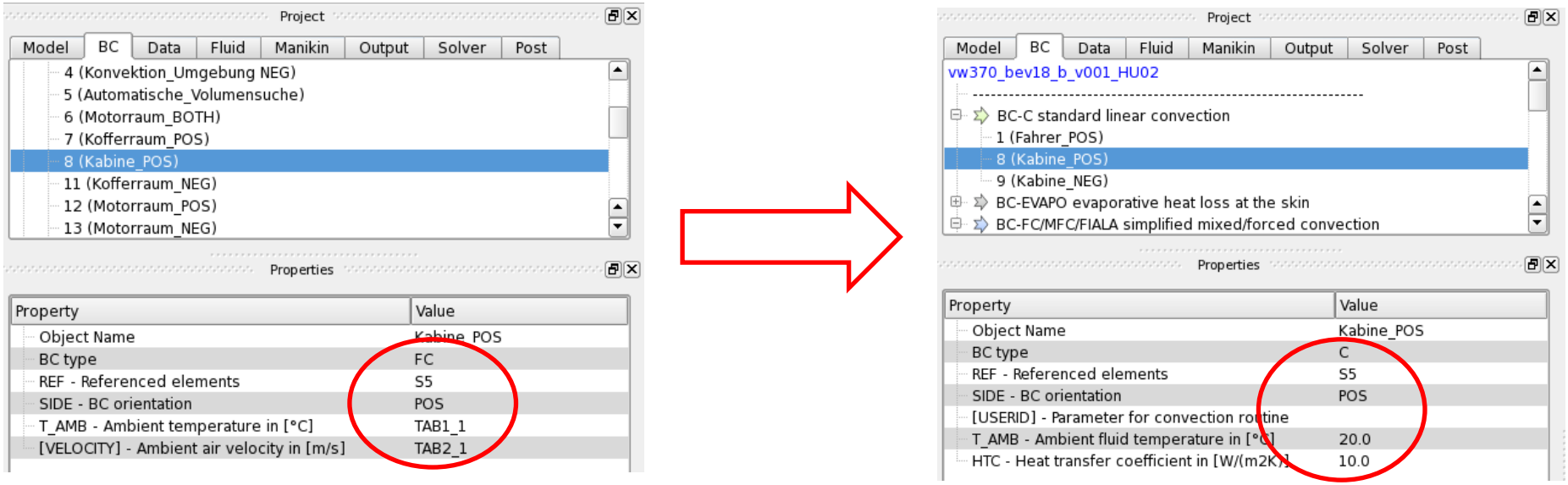


Checklist for co-simulation tasks

- ✓ Create THESEUS-FE and CFD model.
- ✓ Test both models in “standalone modus”, running independently:
  - THESEUS-FE can use dummy convection parameters, CFD can use dummy wall temperatures
  - Any convergence problems with these independent test models must be dealt with now – co-simulation will not work otherwise!
- ✓ Open THESEUS-FE case in the GUI. Start the Coupler Setup Dialog.
  - Define communication mode (file-based or via TCP/IP)
- ✓ Start solver processes and coupler process (in any order). Coupler process will take control of the solver processes
- ✓ Start the Coupler Monitor in the THESEUS-FE GUI to control the running co-simulation task.
- ✓ Post-processing .
- ✓ Possibly more simulations

Preparation of the THESEUS-FE case

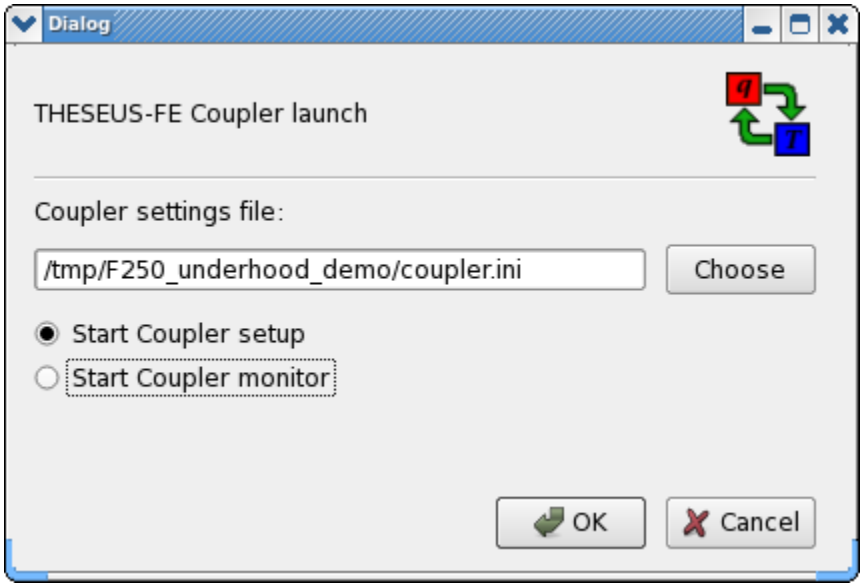
- Replace any AIRZONE objects – convection in fluid regions will now be treated by the CFD code.
- BC objects describing the convection between solid parts and fluid regions need to be changed:



- An initial condition with CONVT=TEMPE avoids extremely large heat fluxes at the beginning of the simulation.

Coupler Launch Dialog

- Load the standalone THESEUS-FE case into GUI.
- The Coupler launch dialog will guide the user through all necessary preparation steps:



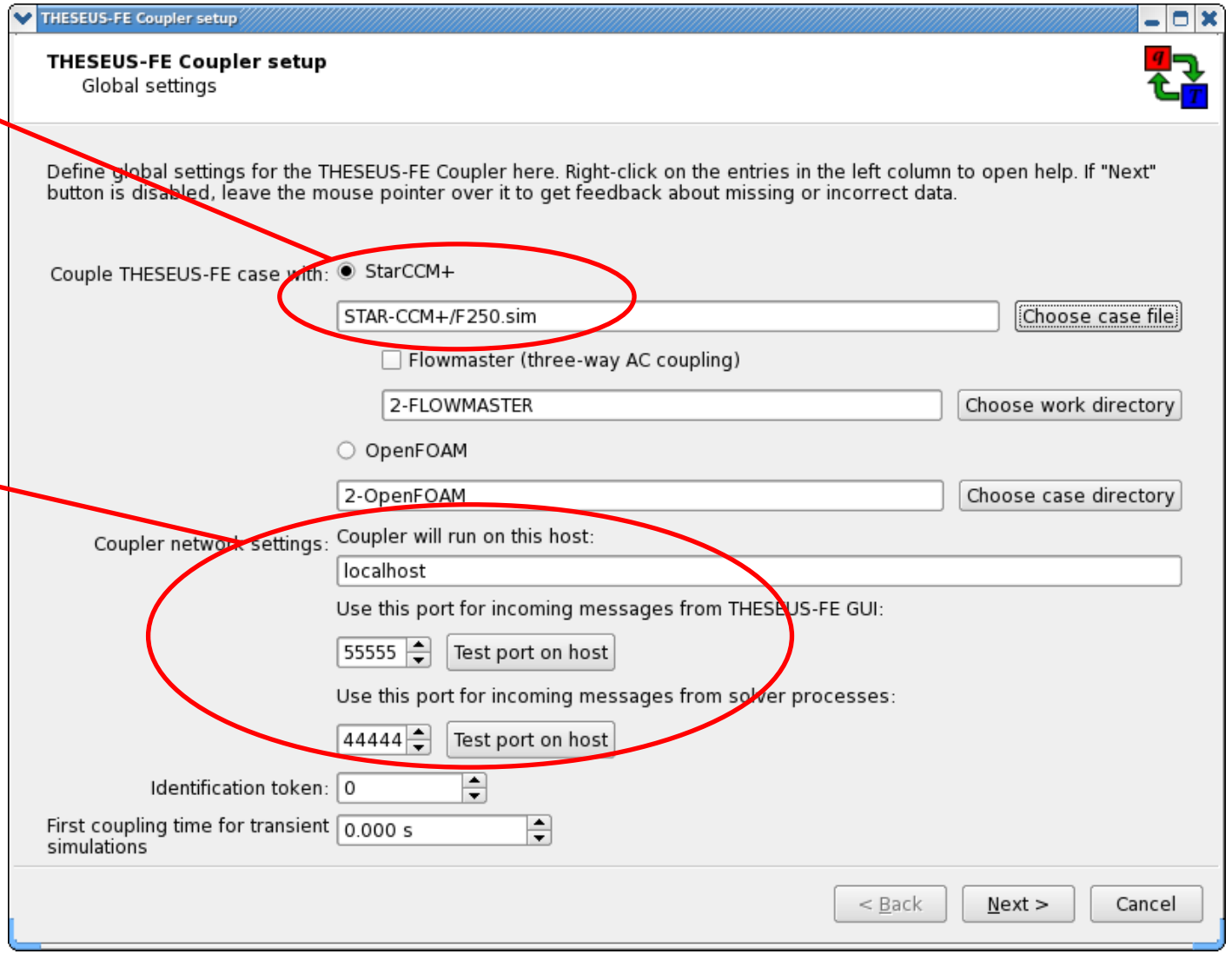
- First select option “Start Coupler Setup”.
- The Coupler working directory will later contain a file „coupler.ini“ with all selection coupling options. It is a good idea to put the THESEUS-FE and CFD solver files in sub-directories of the working directory.

Coupler Setup Dialog Page 1

- Page 1 of Coupler Setup Dialog: global configuration

Select Star-CCM+ and the .sim file

Settings for TCP/IP-communikation: Where will the Coupler process run, and how will you connect to it?



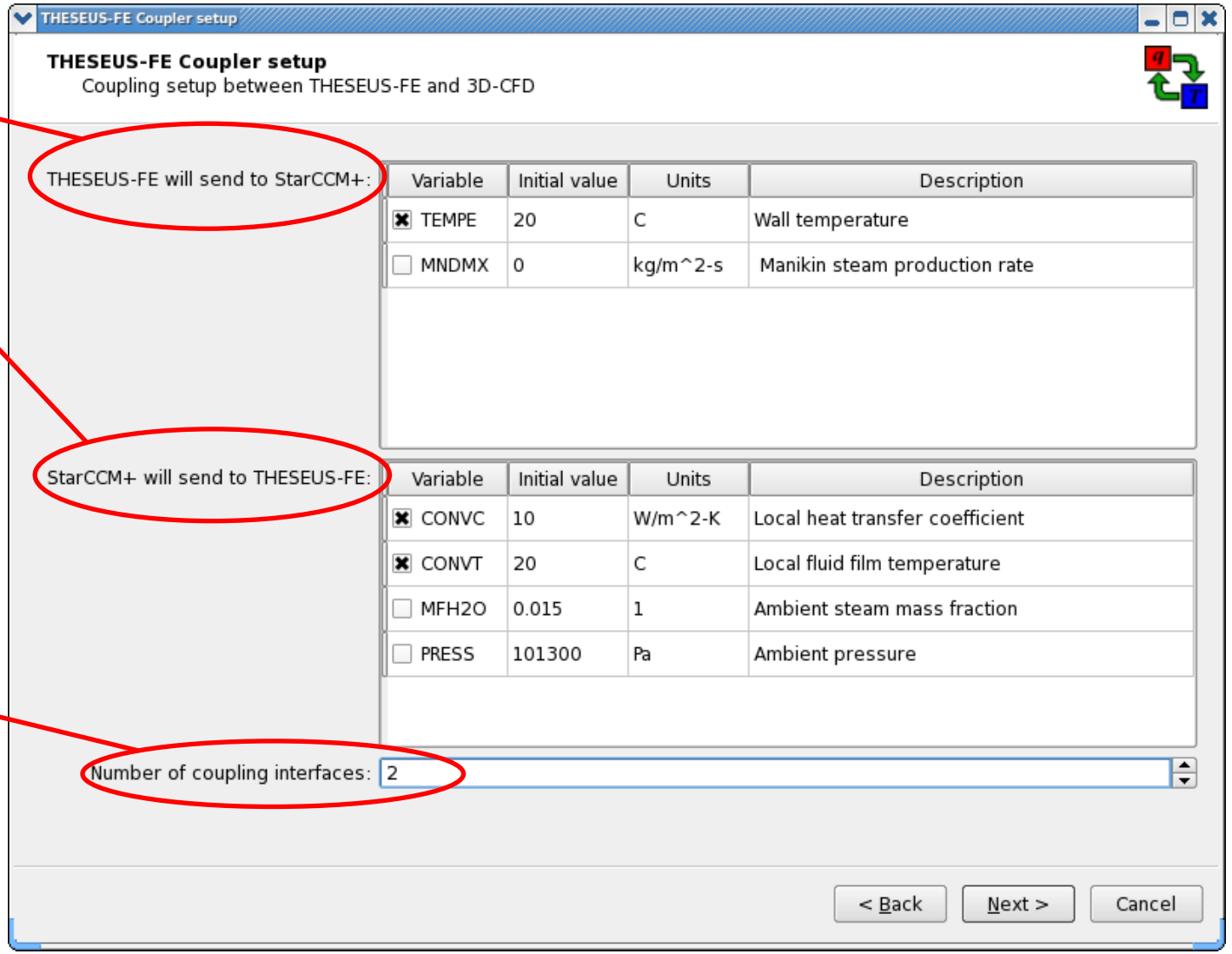
Coupler Setup Dialog Page 2

- Page 2 of Coupler Setup Dialog: exchanged quantities:

THESEUS-FE sends wall temperature

Star-CCM+ sends α und T_{fluid}

Number of coupling interfaces, see next page



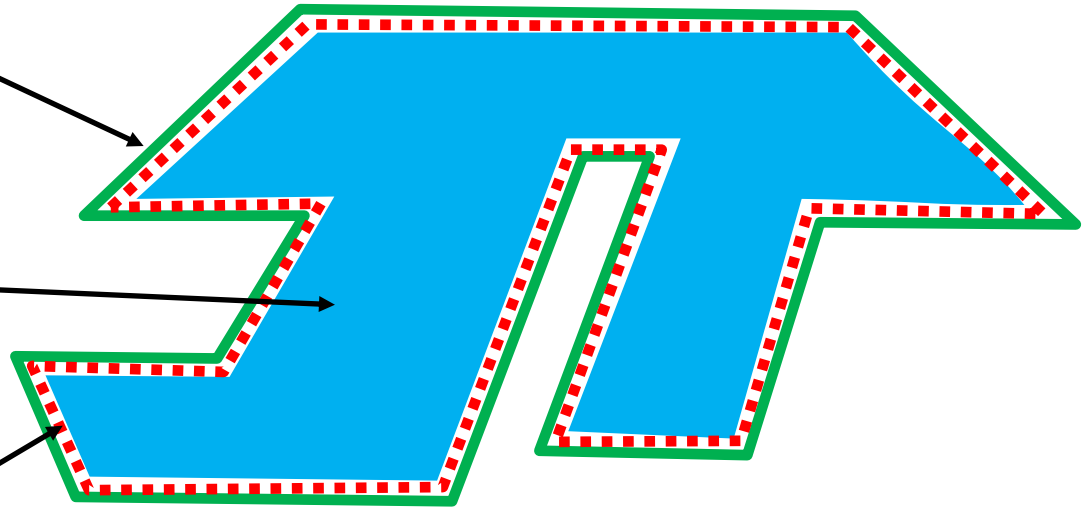
Explanation of coupling interfaces

An **Interface** is the boundary between the physical domains treated by the individual solvers.

Solid parts: THESEUS-FE

Air: CFD

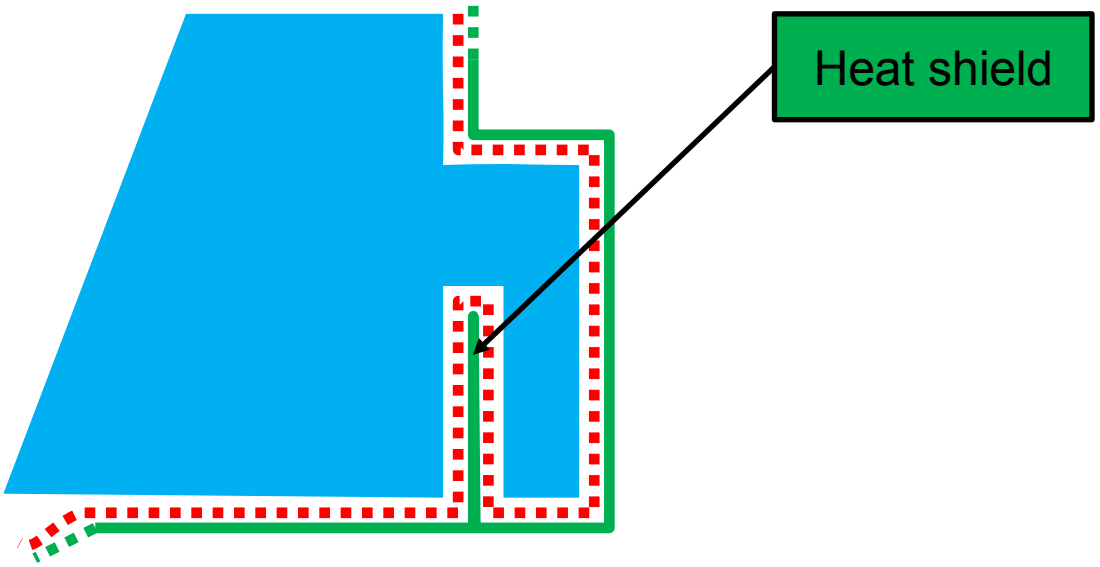
Interface between solid parts and air



The meshes used for the discretization of solid parts and air volume will not be conformal in general. Any data crossing the interface must therefore be interpolated (“mapped”). This task is carried out by the THESEUS-FE Transformer tool.

More on interfaces

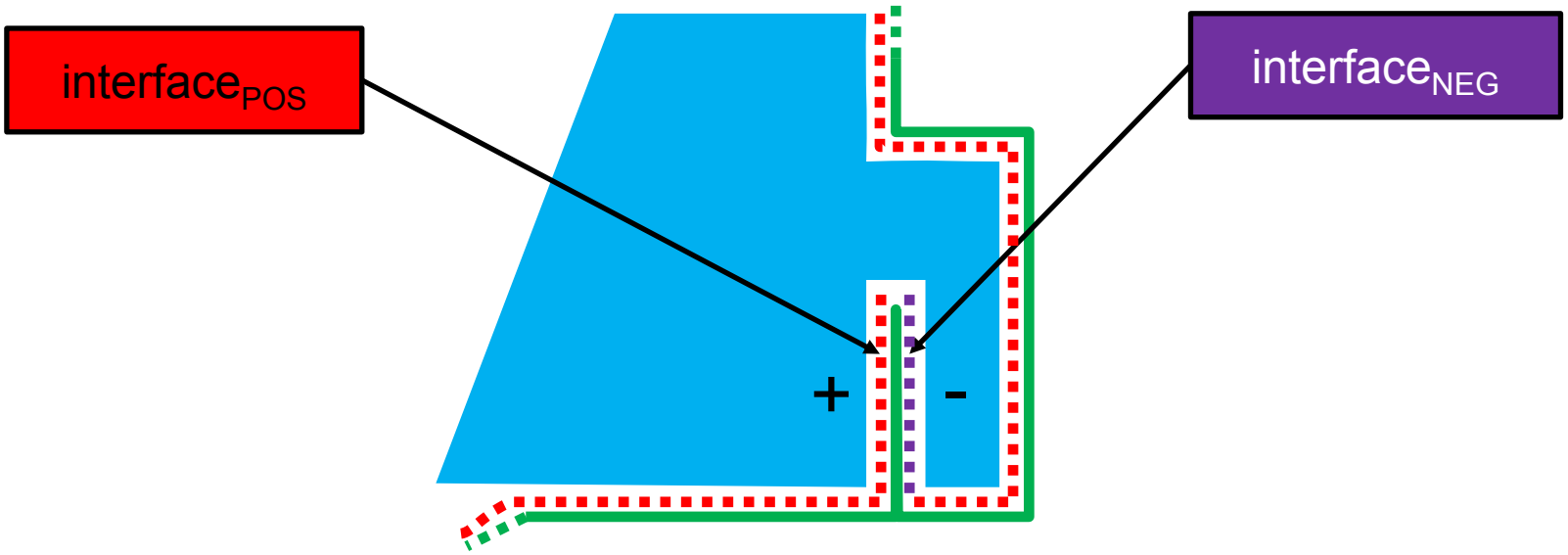
Thin-wall parts, e.g. a heat shield, will usually be modelled using a shell layer in THESEUS-FE. Both sides of the wall are in contact with air:



With „naive“ mapping of α und T_{fluid} one risks incorrect results since the Transformer cannot distinguish between front and back of a shell surface – front and back shell surface are at the same geometric position. The mapped T_{fluid} field will be a random mix of front and back values.

Even more on interfaces

Our solution for this problem: the use must define two separate interfaces. Data mapping is carried out independently for each interface:



interface$_{POS}$ and interface$_{NEG}$ are assigned to the positive respective negative shell side in THESEUS-FE. The Transformer will now map convection values from adjoining fluid volume elements to the correct shell side.

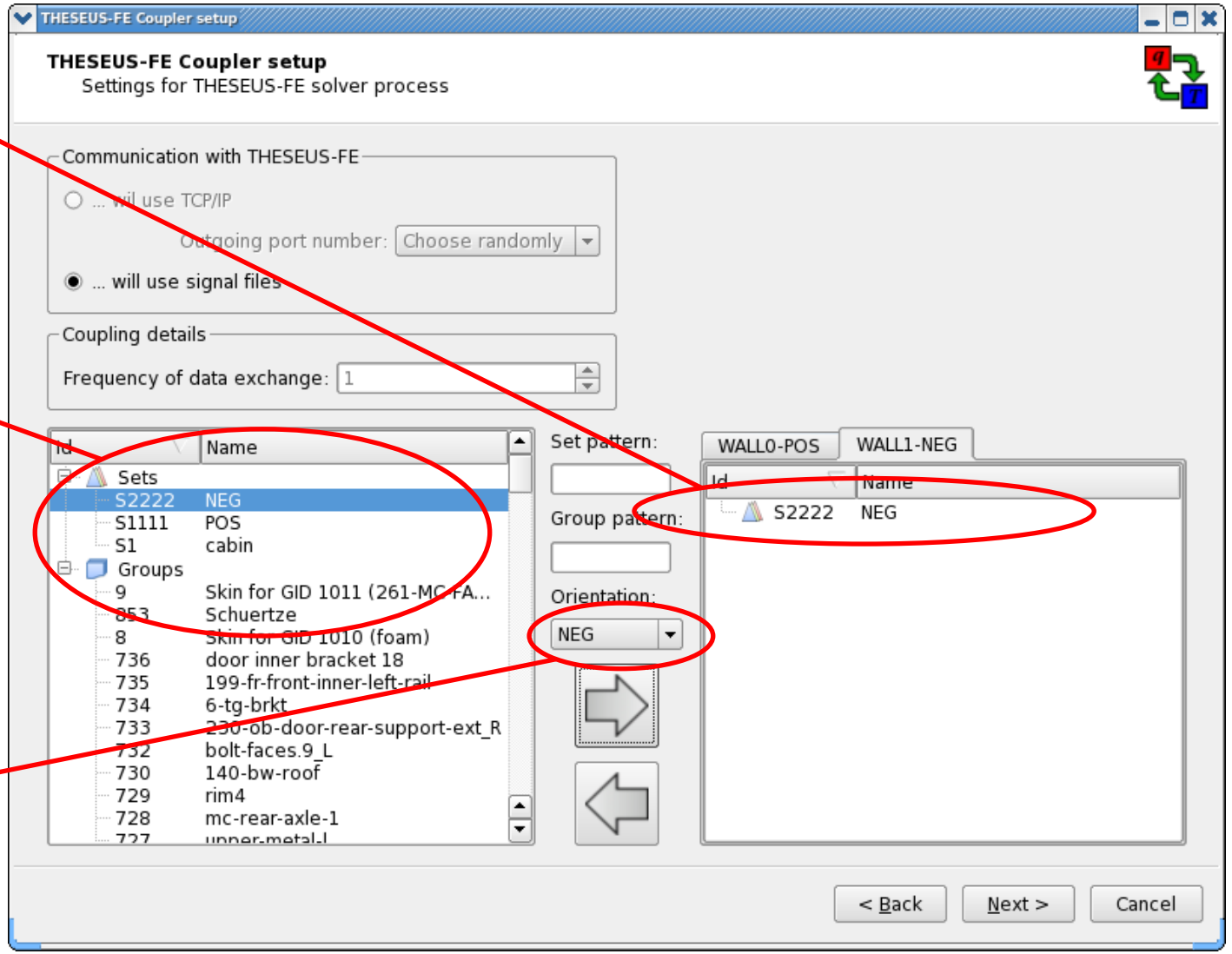
Coupler Setup Dialog Page 3

Page 3 of the Coupler Setup Dialog: interface definition for THESEUS-FE:

One of more SETs and/or PIDs can be used to define the interface

All SETs and PIDs occuring in the case

Select the correct orientation for the interface here!



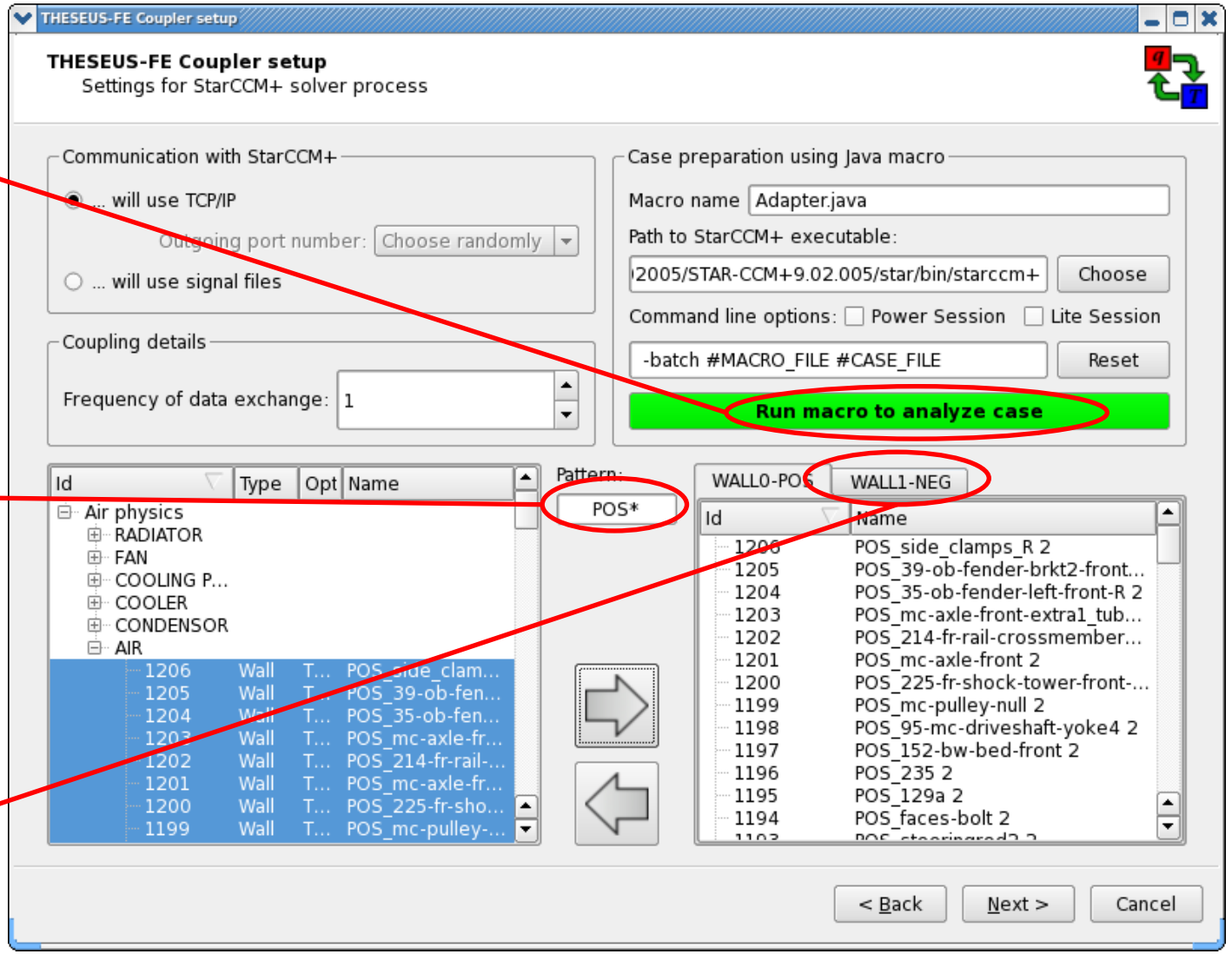
Coupler Setup Dialog Page 4

Page 4 of the Coupler Setup Dialog: Interface definition for Star-CCM+:

This button starts a Java macro to analyze the .sim file. Click this first!

Use the pattern selection helper to select all Star-CCM+ boundaries whose name starts with "POS"

Repeat this step for negative interface



Coupler Setup Dialog Page 5

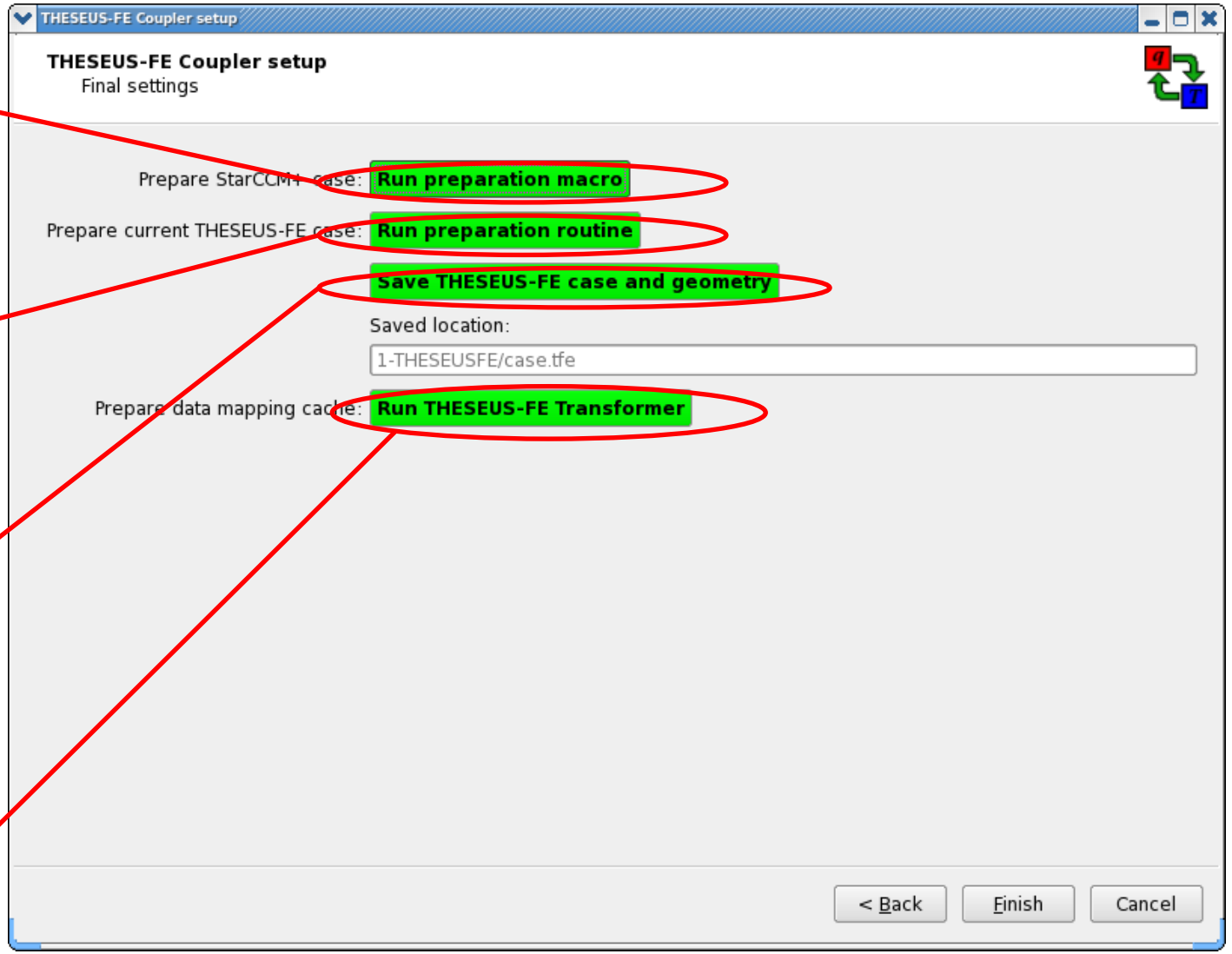
Page 5 of Coupler Setup Dialog: Concluding steps

Finalize Star-CCM+ settings

Finalize THESEUS-FE settings

Save THESEUS-FE case

Create interpolation matrix for Transformer



Coupler Monitor

The Coupler monitor is used to control the co-simulation.

Coupler controls

Log window